

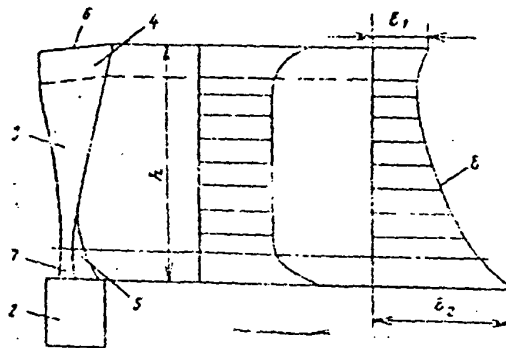
ZHUR/ ★ Q56 K5257B/45 ★ SU-646-09:
Axial compressor working blade - has specified periphery and base
profile with curve angle increasing towards blade end

ZHURALEV YU I 21.09.77-SU-525729

(08.02.79) F04d-29/38

The compressors blade's root and the periphery curves
angle is gradually imcreased towards the ends in order

to make the work more
economical.



The blade which
consists of the feather
(1) and the tail (2) has
a part (3) of the
diminishing angle of
curved profile rela-
tive to the theoretical
pressure line. The
blades periphery and
the root parts (4) and
(5) which represent
10-15% of its height

"h" - are made with the gradually increasing curve angle
reaching at the ends (6) and (7) valves E1 and E2 which
exceed the angle "E" by 10-20%.

In the multistage design, the additional energy expended
on the parts (4) and (5) results in the uniform speed
downstream of the wheel and the reduced angle of entry to
the next stage. Zhuravlev Yu. I., Revzin B. S.,
Tarabrin A. P. Bul. 5/5.2: 79 21.9.77 as 525729 (2pp121)

значения угла изгиба ϵ на границах участка 3 на $10 \div 20\%$.

При работе лопатки в состав ступени компрессора дополнительный подвод энергии в периферийной и корневой частях 4 и 5 лопатки приводит к выравниванию поля скоростей за рабочим колесом и к уменьшению углов атаки на входе в последующую ступень, что обеспечит повышение экономичности.

Описанное техническое решение целесообразно применять для предвключенных ступеней, размещаемых перед моделируемым отсеком, что характерно при проектировании нового компрессора на базе исходного отработанного компрессора.

В этом случае постановка предвключенных ступеней мало отразится на эторах скоростей на входе в моделируемый отсек, что позволит также сократить сроки проектирования нового компрессора.

Формула изобретения

Рабочая лопатка осевого компрессора, имеющая участок с уменьшающимся по высоте углом изгиба профиля в соответствии с условием постоянства теоретического напора, отличающаяся тем, что, с целью повышения экономичности, периферийная и корневая части лопатки, составляющие $10 \div 15\%$ от высоты последней, выполнены с монотонно увеличивающимся к концам на $10 \div 20\%$ углом изгиба.

Источники информации, принятые во внимание при экспертизе

1. К. В. Холщевников. Теория и расчет авиационных лопаточных машин. М. Машиностроение, 1970, с. 190-204.

2. Энергетическое машиностроение серия А, № 3, изд. Иностранная литература, 1961, с. 112, 116.

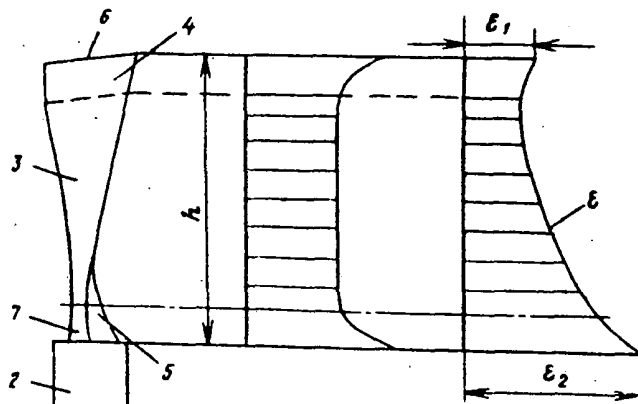


Fig. 1

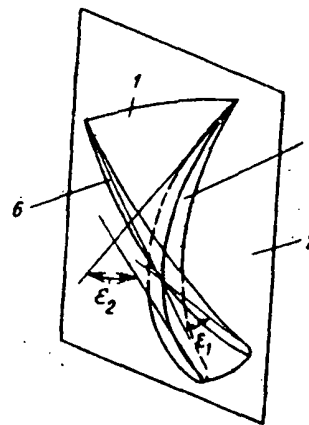


Fig. 2

416/
223A, 228

Редактор В. Большакова Составитель С. Зарцкий
Техред Л. Алферова Корректор П. Макаревич

Заказ 80/26

Тираж 771

Подписное

ЦНИИПИ Государственного комитета СССР
по делам изобретений и открытий

113035, Москва, Ж-35, Раушская наб., д. 4/5

Филиал ППП "Патент", г. Ужгород, ул. Проектная, 4

(54) RUNNER FOR HYDRAULIC MACHINE

(11) 5-272444 (A) (43) 19.10.1993 (19) JP

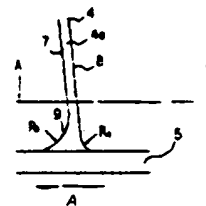
(21) Appl. No. 4-68618 (22) 26.3.1992

(71) TOSHIBA CORP (72) KOICHIRO SHIMIZU

(51) Int. Cl. F03B3/12, F03B11/04

PURPOSE: To suppress vortex cavitation generated at a corner part between the band running water surface and the negative pressure face side of a runner vane inlet tip part even in a high-speed compact hydraulic machine.

CONSTITUTION: In the inlet tip part 4a of a runner vane 4 in a Francis turbine, an angle formed by the negative pressure face side 7 of the runner vane 4 and the band running water surface 5a is an acute angle smaller than 90° . The curved face of a root part 9 between the vane 4 and the band running water surface 5a is determined in such a way that the radius-of-curvature Rb of the vane inlet tip part 4a of the negative pressure face 7 is sufficiently larger than the radius-of-curvature Ra of the other parts. This radius-of-curvature Rb is obtained from experimentation and is 8mm or more at the time of converting the runner outlet diameter into 350mm.



A: rotating direction

(19) 日本国特許庁 (J P)

(12) 公開特許公報 (A)

(11) 特許出願公開番号

特開平5-272444

(43) 公開日 平成5年(1993)10月19日

(51) Int.Cl.³

F 0 3 B 3/12

11/04

識別記号

庁内整理番号

F I

技術表示箇所

7361-3H

7361-3H

審査請求 未請求 請求項の数2(全5頁)

(21) 出願番号 特願平4-68618

(22) 出願日 平成4年(1992)3月26日

(71) 出願人 000003078

株式会社東芝

神奈川県川崎市幸区堀川町72番地

(72) 発明者 清水 光一郎

神奈川県横浜市鶴見区末広町2丁目4番地

株式会社東芝京浜事業所内

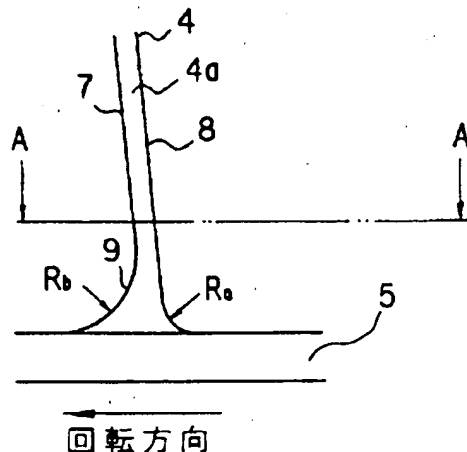
(74) 代理人 弁理士 佐藤 一雄 (外3名)

(54) 【発明の名称】 水力機械のランナ

(57) 【要約】

【目的】 高速化及び小型化された水力機械についても、ランナ羽根入口先端部の負圧面側とバンド流水面とのコーナー部で発生する渦キャビテーションを抑制することができる。

【構成】 フランシス水車のランナ羽根4の入口先端部4aは、ランナ羽根4の負圧面側7とバンド流水面5aとのなす角度が 90° より小さい鋭角である。羽根4とバンド流水面5aとの付け根部9の曲面は、負圧面7の羽根入口先端部4aの曲率半径 R_b がその他の部分の曲率半径 R_a よりも充分に大きくなるように定められている。この曲率半径 R_b は、実験から求められ、ランナ出口径を350mmに換算した時に8mm以上である。



【特許請求の範囲】

【請求項1】周方向に等間隔に配置された複数のランナ羽根と、これらのランナ羽根の外周部を互いに固定するランナバンドとを具備し、上記ランナ羽根の負圧面側と上記ランナバンドの流水面との付け根部が所定の曲率半径の曲面によって形成されている水力機械のランナにおいて、上記曲面は、上記ランナ羽根の流体入口先端部での曲率半径がランナ出口径を350mmに換算した時に8mm以上であると共に、上記流体入口先端部から下流に向かうに従って曲率半径が小さくなるように、構成されていることを特徴とする水力機械のランナ。

【請求項2】上記ランナ羽根の流体入口先端部の付近における上記ランナ羽根の負圧面は、上記ランナバンド流水面から大きく離れた遠方領域と、上記ランナバンド流水面に接近しかつ上記遠方領域に隣接した近傍領域とを少なくとも有し、上記遠方領域は上記ランナバンド流水面とのなす角度が90°よりもかなり小さい鋭角であり、上記近傍領域は上記ランナバンド流水面とのなす角度がほぼ90°であり、上記曲率半径8mm以上の曲面は上記近傍領域と上記ランナバンド流水面との間に形成されていることを特徴とする請求項1に記載の水力機械のランナ。

【発明の詳細な説明】

【0001】

【産業上の利用分野】本発明はフランス水車などの水力機械のランナに係り、特にランナ羽根の負圧面側でのキャビテーションの発生を防止した水力機械のランナに関する。

【0002】

【従来の技術】図5及び図6は水力発電所で広く使用されている一般的なフランス水車の構造を示したのもで、主軸1に取付けられたランナ2は、クラウン3と、周方向に等間隔に配置された複数枚のランナ羽根4と、これらのランナ羽根4の外周部を互いに固定する環状のランナバンド5とから構成されている。流体がガイドベーン6を介してランナ2に流入しランナ2を回転駆動し、主軸1を介して図示を省略した発電機を回転する。

【0003】図7はランナ羽根4とランナバンド5とを拡大して示したもので、このバンド流水面5aに沿ったランナ羽根4の形状が図8に示されている。ランナ羽根4の入口先端部4aは、図9に示したようにランナバンド側の方がクラウン側よりも回転方向に対して後方に位置しており、ランナ羽根4の負圧面側7とバンド流水面5aとのなす角度が鋭角であり、圧力面側8が鈍角になっている。ランナ羽根4とバンド流水面5aとの境界コーナー部、即ち付け根部9は、図8乃至図10に示したように、先端部4aを含め羽根4のほぼ全周にわたってほぼ同一の曲率半径Raの曲面に形成されている。

【0004】このような構成の水力機械のランナは、ランナ羽根4の入口負圧面とバンド流水面5aとの付け根

部9から渦が発生し、この渦中心の圧力低下に伴い渦キャビテーションが生じる。このような渦キャビテーションは、図5に示したように羽根入口先端部4aの下流に壊食Aを惹起し、この壊食の度合いが進行すると、水車性能を低下させ、ついには水車の運転自体にも支障をきたすことがある。

【0005】そこで、このような渦キャビテーションの発生を抑制するランナ構造が、これまで種々提案されている。例えば、特開昭57-37071号公報には図11に示したようにランナ羽根4の入口先端部4aに切欠き10を刻設したランナが開示され、特開昭57-26277号公報には図12に示したようにランナ羽根負圧面に突起11を突設したランナが開示され、特公平1-29989号公報には図13に示したようにランナ羽根4の入口先端部の羽根倒れ曲線を回転方向に対してバンド5側に前進させたランナが開示されている。更に図14に示したようにランナ入口下流側のバンド5とランナ羽根負圧面側とのコーナーに肉盛部12を形成したランナや、図15に示したようにランナ羽根4の入口先端部に貫通孔13を穿孔したランナや、図16及び図17に示したようにランナ羽根4の入口先端部4aに鈍頭形状部14を形成したランナも提案されている。

【0006】

【発明が解決しようとする課題】ところが、近年、水力機械はコスト低減のために、高速化及び小型化が要求され、この結果、ガイドベーンからランナに流入する流れは非常に複雑になり、上述した従来のランナ構造では渦キャビテーションの発生を防止できない場合があることが判明した。これを詳述すると、ランナ羽根4の付け根部9付近に流入する流れは、水車の運転状態が変わると3次元的に変化し、ランナ羽根4の入口先端部4aは静止場から回転場への境界にあたるため、流れの影響を受け易く、従来のようなランナ羽根4の付け根部9の形状では、ランナ羽根入口先端部4aの負圧面側7とバンド流水面5aとのコーナー部に主流方向の流れとは異なった流れが生成し、これによって、渦キャビテーションが発生してしまうことが判明した。

【0007】そこで、本発明の目的は、高速化及び小型化された水力機械にあってもランナ羽根入口先端部の負圧面側とバンド流水面とのコーナー部における渦キャビテーションの発生を十分に抑制することができる水力機械のランナを提供することにある。

【0008】

【課題を解決するための手段】ランナ羽根の負圧面側とランナバンドの流水面との付け根部の形状が渦キャビテーションの発生に及ぼす影響を測定する実験を行ったところ、ランナ羽根の流体入口先端部における付け根部の曲率半径が、渦キャビテーションの発生に大きな影響を及ぼし、この曲率半径がランナ出口径を350mmに換算した時に8mm以上であれば、渦キャビテーションの

3

発生を十分に抑制できることが実験により判明した。

【0009】そこで、本発明は、周方向に等間隔に配置された複数のランナ羽根と、これらのランナ羽根の外周部を互いに固定するランナバンドとを具備し、上記ランナ羽根の負圧面側と上記ランナバンドの流水面との付け根部が所定の曲率半径の曲面によって形成されている水力機械のランナにおいて、上記曲面は、上記ランナ羽根の流体入口先端部での曲率半径がランナ出口径を350mmに換算した時に8mm以上であると共に、上記流体入口先端部から下流に向かうに従って曲率半径が小さく

なるように、構成されていることを特徴とするものである。

【0010】この構成にあつては、上記ランナ羽根の流体入口先端部の付近における上記ランナ羽根の負圧面は、上記ランナバンド流水面から大きく離れた遠方領域と、上記ランナバンド流水面に接近しかつ上記遠方領域に隣接した近傍領域とを少なくとも有し、上記遠方領域は、上記ランナバンド流水面とのなす角度が90°よりもかなり小さい鋭角であり、上記近傍領域は、上記ランナバンド流水面とのなす角度がほぼ90°であり、上記曲率半径8mm以上の曲面は上記近傍領域と上記ランナバンド流水面との間に形成されていることが望ましい。

【0011】

【作用】ランナ羽根の負圧面側とランナバンドの流水面との付け根部の曲率半径はランナ出口径を350mmに換算した時に8mm以上に定められているため、上記付け根部の曲面形状は従来の鋭角的な曲面に比べて非常に緩い曲面になる。これによって、ランナ羽根に流入する流れが水力機械の運転状態に応じて変化しても、泡を発生することなくスムーズに流れ、キャビテーションの発生を抑制する。

【0012】

【実施例】以下に本発明による水力機械のランナの実施例を図5乃至図17と同部分には同一符号を付して示した図1乃至図4を参照して説明する。図1及び図2において、フランス水車のランナ羽根4の入口先端部4aは、負圧面側7とバンド流水面5aとのなす角度が鋭角になり、圧力面側8が鈍角になっている。ランナ羽根4とバンド流水面5aとの境界コーナー部、即ち付け根部9の曲面は、負圧面7の羽根入口先端部4aの曲率半径Rbがその他の部分の曲率半径Raよりも充分に大きくなるように定められている。

【0013】この先端部負圧面の付け根部9の曲率半径Rbは以下の実験から求めた。ランナ出口が350mmの模型フランス水車を使用して、先端部負圧面の付け根部9の曲率半径Rbを4mm、8mm、10mm、15mmとしてキャビテーションの発生状況を調べた実験結果を図3に示す。

【0014】この図3のグラフは、横軸が曲率半径Rbであり、縦軸は模型の最高効率 η_0 に対する相対等効率

4

η の比であり、縦軸の数値の小さい方は実機の高落差運転に相当し、大きい方は低落差運転に相当する。フランス水車の場合の使用範囲は、最高効率点を中心とした範囲であるが、最高効率点の単位回転速度に相当する落差近傍からそれよりも低落差側の運転が一般的である。従って、縦軸の1.0よりも大きいところでキャビテーションが発生した場合には、実機でも入口側にキャビテーションが発生し、壊食を受ける危険度が大きいことを示している。この実験では、縦軸の効率は曲率半径Rbが約8mm近傍で1であり、かつ8mm以上では1より小さい。

【0015】従って、曲率半径Rbがランナ出口径を350mmに換算した時に8mm以上であれば、運転範囲内においてランナ4の入口先端部付近の付け根部9での渦キャビテーションの発生を充分に抑制することができる。

【0016】図4はランナ羽根4の入口の羽根倒れが大きい場合の本発明の第2の実施例を示したもので、ランナ羽根4はその入口先端部4aの負圧面7とバンド流水面5aとのなす角度 θ が90°よりもかなり小さな鋭角に形成されている。このような場合には、入口羽根負圧側の入口倒れ曲線をランナ羽根4の負圧面7の途中からバンド流水面5aに垂直またはそれに近くなるように補正した後に、付け根部9の曲率半径をRbとする。

【0017】これを詳述すると、ランナ羽根4の負圧面7を、ランナバンド流水面5aから大きく離れた遠方領域7aと、ランナバンド上端5aに接近しかつ遠方領域7aに隣接した近傍領域7bとに分ける。この時、遠方領域7aはランナバンド流水面端5aとのなす角度 θ がかなり小さな鋭角である。他方、近傍領域7bはバンド流水面5aとのなす角度が90°、またはそれに近い角度であり、上記曲率半径8mm以上の曲面Rbは近傍領域7bとランナバンド流水面5aとの間に形成される。これによって、負圧面7が比較的小さな鋭角にも拘らず、ランナ4の入口先端部付近の付け根部9での渦キャビテーションの発生を充分に抑制することができる。

【0018】

【発明の効果】以上の説明から明らかなように本発明によれば、ランナ羽根の負圧面側とランナバンド流水面との付け根部の曲面は、ランナ羽根の流体入口先端部での曲率半径がランナ出口径を350mmに換算した時に8mm以上となるように構成されているため、泡キャビテーションの発生を充分に抑制することができる。

【図面の簡単な説明】

【図1】本発明による水力機械のランナの第1の実施例のランナ入口先端部とバンド流水面とを示した概略図。

【図2】上記実施例の付け根部の等高線を示した図1のA-A線に沿った断面図。

【図3】上記実施例の付け根部の曲率半径と水車効率との関係を求めた実験結果を表すグラフ。

5

6

【図4】本発明による水力機械のランナの第2の実施例のランナ入口先端部とバンド流水面とを示した概略図。

【図5】一般のフランシス水車の構造を示した断面図。

【図6】上記一般のフランシス水車のランナを示した斜視図。

【図7】図5のランナバンドを拡大して示した断面図。

【図8】図7のバンド流水面に沿ったランナ羽根の形状を示した断面図。

【図9】図5のランナのランナ入口先端部とバンド流水面とを示した概略図。

【図10】付け根部の等高線を表す図9のB-B線に沿った断面図。

【図11】従来のランナを示した断面図。

【図12】別の従来のランナを示した断面図。

【図13】別の従来のランナを示した斜視図。

【図14】別の従来のランナを示した断面図。

【図15】別の従来のランナを示した断面図。

【図16】別の従来のランナを示した断面図。

【図17】図16のC-C線に沿ったランナ羽根の形状を示した断面図。

【符号の説明】

2 ランナ

4 ランナ羽根

4a 入口先端部

10 5 ランナバンド

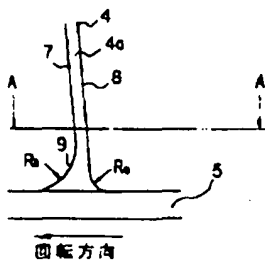
5a ランナバンド流水面

7 負圧面

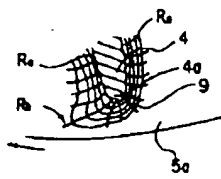
9 付け根部

Rb 曲率半径

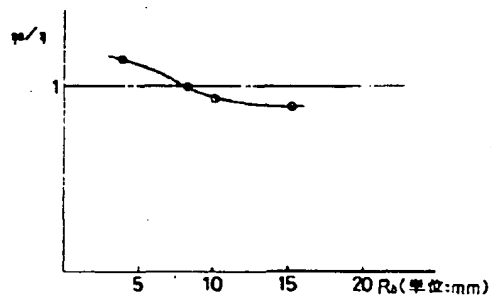
【図1】



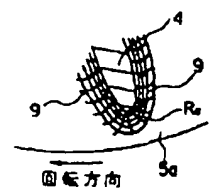
【図2】



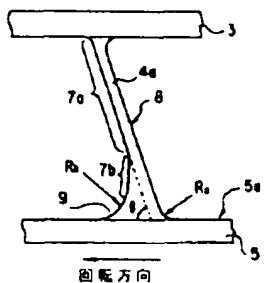
【図3】



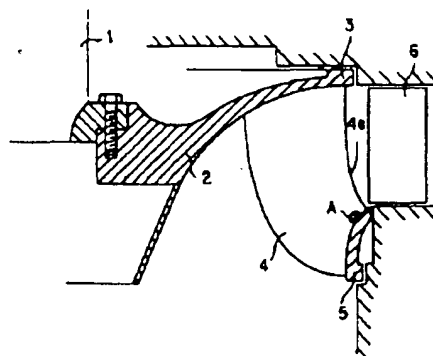
【図10】



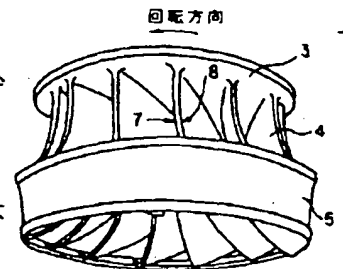
【図4】



【図5】

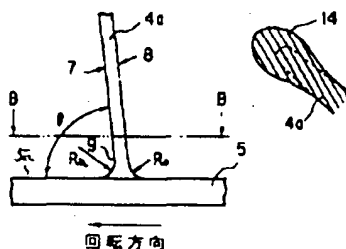


【図6】

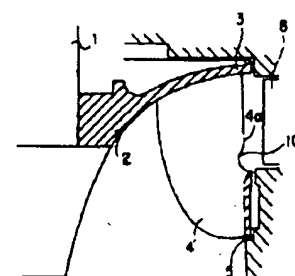


【図9】

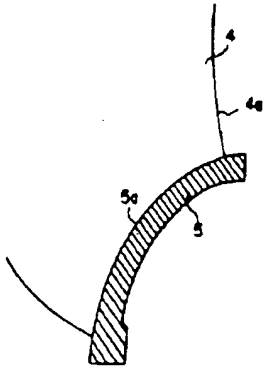
【図17】



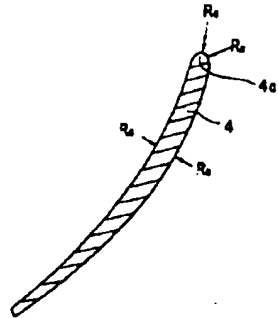
【図11】



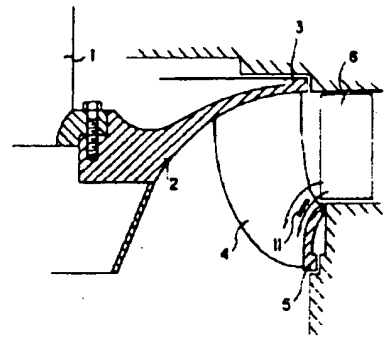
【図7】



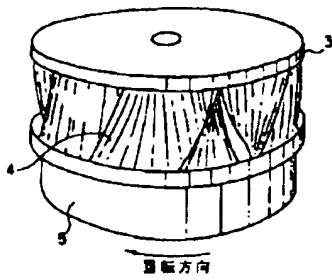
【図8】



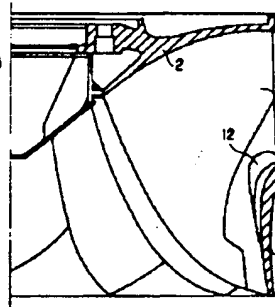
【図12】



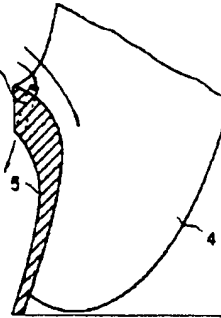
【図13】



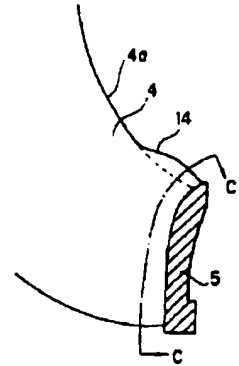
【図14】



【図15】



【図16】



(54) TURBINE VANE TRAIN(11) Kokai No. 52-74706 (43) 6.23.1977 (21) Appl. No. 50-150597

(22) 12.19.1975

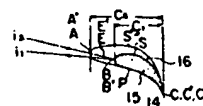
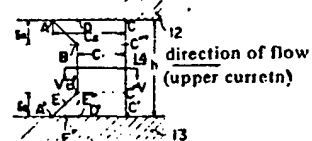
(71) HITACHI SEISAKUSHO K.K. (72) TAKESHI SATO

(52) JPC: 50B92;51B731

(51) Int. Cl². F01D5/14

PURPOSE: To prevent a lowering of a turbine efficiency by preventing a generation of a secondary flowing between turbine vanes.

CONSTITUTION: A vane 14 bordered by both upper and lower side walls 12, 13 within a definite length has an increasing vane width gradually from C_1 to C_2 accompanied with approaching to side walls 12, 13 within a range of δ_2 in the vicinity of side walls 12, 13. i.e. Against for a vane width C_1 of a former vane DCC'D', formed up to enlarge in the upper current side within a range of δ_2 respectively as ABD at the vicinity of an upper side wall 12 and as A'B'D' at the vicinity of a lower side wall 13. A vane form of an enlarged vane width as such has a coincided front surface P of within a range of a vane width C_1 and C_2 , however back surfaces S, S' are increasing their vane thicknesses from back surfaces S to S' accompanied with increasing of a vane width from C_1 to C_2 . For this sake a generation of a border layer between vanes, i.e. a secondary flow can be prevented.





特許願なし

(特許法第38条九款し書)
の規定による特許出願

特 許 願 (4)

(4,000円)

昭和 50 年 12 月 19 日

特許庁長官 斎藤 英 雄 殿

1. 発明の名称 タービン翼列

1. 特許請求の範囲に記載された発明の枚数 3

2. 発明者 ヒダシタイワイワ
茨城県日立市津町3丁目1番1号
株式会社日立製作所日立研究所内
氏名 佐藤 武

3. 特許出願人
住所 東京都千代田区丸の内一丁目5番1号
氏名 (510) 株式会社日立製作所
代表者 吉山 博吉

4. 代理人
住所 〒105 東京都港区有明1丁目6番11号 ティロイトビル
氏名 (5926) 秋本 正実
電話東京 (591) 4414番

5. 添付書類の目録

- (1) 明 細 書 1通
- (2) 図 面 1通
- (3) 願 書 副 本 1通
- (4) 全 体 50頁 150597

分式 (5)

① 日本国特許庁

公開特許公報

①特開昭 52-74706

④公開日 昭52.(1977) 6.23

②特願昭 50-150597

②出願日 昭50.(1975) 12.19

審査請求 未請求 (全4頁)

庁内整理番号

7526 32
6620 32

⑤日本分類

50 B92
51 B73/

⑤ Int. Cl?

F01D 5/14

識別
記号

明 細 書

発明の名称 タービン翼列

特許請求の範囲

- (1) 翼の先端と根元あるいは翼の根元に隔壁を有する翼を配列して限定された流路を構成するタービン翼列において、前記各翼の弦長を隔壁からの所定範囲内で隔壁に接近するに伴い上流側に向つて漸次拡大するように構成したことを特徴とするタービン翼列。
- (2) 特許請求の範囲第1項記載のタービン翼列における弦長の拡大された範囲の各翼の断面形状を、弦長の拡大に伴つて翼の背面側の凸部を増大して厚肉形状にしたことを特徴とするタービン翼列。
- (3) 特許請求の範囲第1項記載のタービン翼列における弦長の拡大された範囲の各翼の断面形状を、弦長の拡大に伴つて翼の背面側の凸部を増大して厚肉形状にすると共に、翼の上流部分における流入角度を背面側に傾斜するようにしたことを特徴とするタービン翼列。

発明の詳細な説明

本発明は翼の先端と根元および根元のみに隔壁を有する翼を限定された流路に配列してなるタービン翼列に関するものである。

タービンの翼はその翼長が有限であるため、必ず隔壁が設けられている。この隔壁は境界層を発達させ、また二次流れを発生させるので、タービンの性能を劣化させることは周知である。この性能劣化は翼長の短いほど大であり、蒸気タービンでは高、中圧段落における段落効率を左右するところ大である。

従来のタービン翼列における隔壁近傍の流を第1図および第2図について説明する。

第1図において、隔壁2に直交する翼1、1'間の流路3における流れは、隔壁2から十分に離れた位置では流線4aのように流路3に沿つたものとなり、隔壁2に接近すると隔壁2に発達した境界層との干渉により、通常の境界層速度分布に横断速度分布Wが付加され、流線4aの流れ角 β_1 と異なつた流れ角 β_2 の流線4bのようになる。このため

異間流路内におけるエネルギー損失分布は第3図(1)に示すとおりで、異長の中央部における損失は流線4bによる損失10のみであるため比較的小さいが、側壁2の近傍のb範囲では前記8、9のために損失11(二次流れ損失)は非常に大となる。

このように異幅を拡大した翼の形状は第5図に示すように、異幅 c_1 および異幅 c_2 の範囲の腹面Pは一致するが、背面S、 δ' は異幅が c_1 から c_2 に増加に伴って翼厚は背面Sから背面 δ' まで増加する。

図 14a, 14b の腹面 P および背面 S, b の分布は第 7 図に示すとおりである。すなわち魚形 15, 16 の腹面 P の形状は相似であるため、実線 17 で示すように同一になる。また第 6 図より明らかなように風向流路が魚形 15 では腹面 P と背面 S で形成されるに対し、魚形 16 では腹面 P と背面 G で形成

上記のような圧力の傾向により翼長方向の流線の変化は第8図のようになる。すなわち第6図に示した翼列構成によると、背面側において上流側から流入する各流線 $\phi_1 \sim \phi_2$ のうち流線 $\phi_1 \sim \phi_2$ は第6図の翼形拡張部に相当する β/β_0 （第8図）の範囲において直進することなく圧力の低い側壁13の方向に曲げられ下流側では流線 $\phi_3 \sim \phi_4$ となる。

- これは従来の翼形1と同形の翼15(第8図のB'でDの範囲)において、第2図の渦8のローアップと渦7の流れにより生ずる翼長の中央部に
 1 向う流れ例えば第8図の渦線 $\phi_1 - \phi_2$ 、 $\phi_2 - \phi_3$ および $\phi_3 - \phi_4$ と逆の作用をすることになる。したがって側壁近くの翼形の背面で発生する渦のローアップが減少されると共に、第7図の圧力分布
 5 からわかるように側壁近くの流体は従来の翼形よりも加速されるため、第9図に示す側壁の流れ境界層の横断速度成分 w は減少し、側壁上の流れは従来のものでは $\phi' - \phi_1'$ に対し、本発明の翼形では
 10 同流線は $\phi' - \phi_1'$ となり側壁の影響を受けない流線 $\phi' - \phi_1'$ に接近する。よって本発明によれば側壁近くのエネルギー損失を低減させることができる。
 15 多段タービンの場合、上流の翼列から流出する流れの翼長中央部における速度三角形は第10図に示すように U_1 、 w_1 および C_1 で形成されている。そして側壁に近づくに伴って第9図に示すように、
 20 境界層内では速度 U は $U_1 > U_2 > U_3$ のように次第に減少し、タービンの回転周速成分 w も $w_1 > w_2 > w_3$

のように減少するが、速度 U の変化が大であるため、次段への流入方向は側壁に近づくに伴って流入角が ϕ_1 から ϕ_3 のように大となる。したがって翼形15(第6図)の形状で側壁まで形成されている翼列においては、側壁近くでは背面(8面側)の流入角は ϕ ではなく ϕ' となる。このように翼形15の流入角 ϕ が実際の流入角 ϕ' にすれば翼面の流れにはくねりなどを生じ、側壁近くでは前記二次流れと干渉して著しく流れ損失が増大し翼列の性能を低下させる。

しかるに本発明では各翼の弦長が側壁からの所定範囲内で側壁に接近するに伴い上流側に向って漸次拡大するようにし、かつこの拡大に伴って翼の背面の凸部を増大して厚肉形状に成形する、すなわち第4図～第6図に示すように側壁からも、範囲で側壁に近づくに伴って第10図に示す速度三角形の変化を考慮して翼形の流入角を ϕ から ϕ' に滑らかに変化させ、流入角の変化による損失をなくするように構成したので、翼列の性能低下を防止することができる。

- 以上説明したように、本発明によれば側壁近くで発生する二次流れ損失および側壁近くの流入角の変化により発生する翼形損失を大幅に低減し、翼列の性能を向上させることができる。

5 図面の簡単な説明

- 第1図は従来のタービン翼列の翼間流路における流れの説明図、第2図は同側壁近くの流れ説明図、第3図(ハ)は同翼間流路内のエネルギー損失分布図、第3図(ヘ)は同図(ハ)のⅢ-Ⅲ線における断面図、
 10 第4図は本発明のタービン翼列の一実施例を示す構成説明図、第5図は第4図のⅣ-Ⅳ線における断面図、第6図は本発明のタービン翼列を示す拡大詳細図、第7図は本発明における翼面の圧力分布図、第8図は本発明における翼長方向の流れの変化を示す説明図、第9図は本発明における側壁の境界層の速度分布図、第10図は本発明における翼形の速度三角形の説明図である。

符 号 の 説 明

- 12, 13 側壁
 20 14 翼

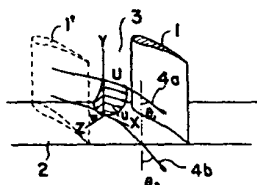
P	背面
S, S'	腹面
ϕ, ϕ'	流入角

特許出願人 株式会社 日立製作所
 代理人 井堀士 秋 本 正 実

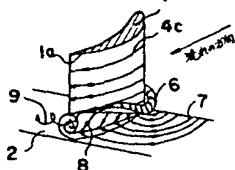
JAPAN (6-1977)

REF ID: A6252-74706-40

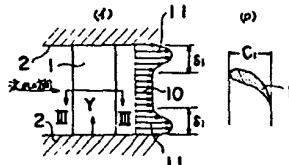
214



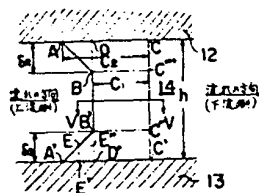
第 2 圖、



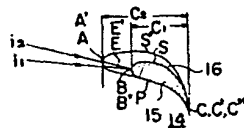
第 3 回



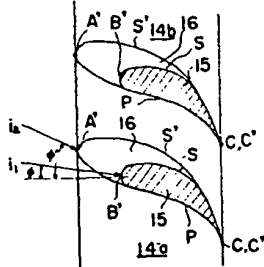
第 4 章



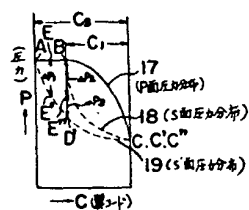
第 5 题



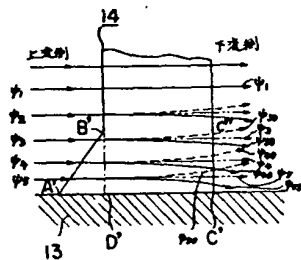
第 6 回



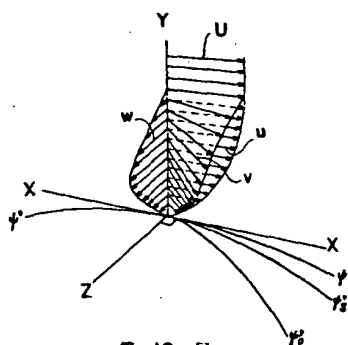
第 7 圖



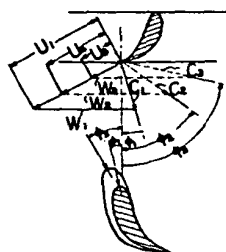
第 8 圖



第 9 题



第 10 章

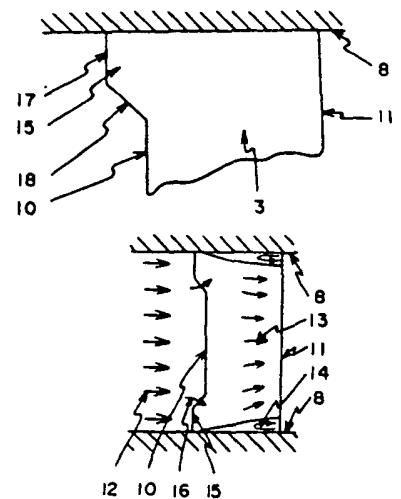

$$\begin{array}{r} 4.6 \\ \hline 223.9 \\ 4.5 \\ \hline \text{Diff. } 1 \end{array}$$

(54) **BLADE IN RETARDATION CASCADE FOR AXIAL-FLOW ROTARY MACHINE**

(11) 59-115500 (A) (43) 3.7.1984 (19) JP
(21) Appl. No. 57-224039 (22) 22.12.1982
(71) JIYUNJI UEMATSU (72) JIYUNJI UEMATSU
(51) Int. Cl. F04D29/54

PURPOSE: To aim at an improvement of efficiency and a reduction in noise, by installing a projection in a part in close vicinity to a wall surface at the inflow end of a conventional retardation cascade, while making part of a main flow turn to a wall surface direction flow, reducing the secondary flow to some extent, and expanding a stall limit of the cascade.

CONSTITUTION: Installing a projection 15 in a blade inflow end 10 nearby a blade wall surface 8, part of a main flow 13 is made to turn to a wall surface direction flow 16, with the effect of which a secondary flow 14 is reduced. In this connection, when the periphery of this projection 15 is expanded wider, as shown in the following illustration, a retardation effect in the secondary flow 14 can be further increased when a taper part 18 connecting a part 17 being almost paralleled with the blade inflow end 10 and the latter 10 is installed. Doing like this, the secondary flow 14 is reduced whereby an improvement of efficiency is well promoted and, what is more, such a noise as caused by whistling action at a blade outflow end 11 is also reduced as a flow velocity of the main flow 13 at the blade outflow end 11 is reduced.



⑤ Int. Cl.³
F 04 D 29:54

識別記号

庁内整理番号
7532—3H

⑬ 公開 昭和59年(1984)7月3日

発明の数 1
審査請求 未請求

(全 2 頁)

⑭ 軸流回転機械減速翼列の翼

⑯ 発明者 上松順二
八王子市片倉町1069—182⑰ 特 願 昭57—224039
⑱ 出 願 昭57(1982)12月22日⑲ 出 願 人 上松順二
八王子市片倉町1069—182

明 細 書

1. 発明の名称

軸流回転機械減速翼列の翼

2. 特許請求の範囲

1 翼長手方向にある壁面(8)にそった翼端近く
の翼前縁(4)に突起(5)を取りつけたことを特徴
とする翼。2 翼前縁に取りつけた突起は翼流入端(1)にほ
う平行な部分(17)とこれと翼流入端を結ぶテー
パ部(18)よりなる特許請求の範囲第1項記載の
翼。

3. 発明の詳細な説明

この発明は軸流圧縮機や軸流ポンプ等軸流回転
機械で使用する減速翼列の失速限を上げた翼に関
する。従来圧力上昇を目的とした軸流回転機械では減
速翼列を使用し、翼列に流入する流体速度エネル
ギを翼列通過中に減速して圧力エネルギーに変える
が、周知の通り1段の翼列で失速しないで行し得
る仕事には限界がある。此の限界を上げるために翼の断面形状や翼の配列について改良がなされた
が、翼面と共に流体通路を構成する壁面が先に失
速するため、翼形や翼配列を何のように工夫して
も、翼列の流入及び流出理論流速を各 W_1 及び W_2 と
すると、 $C_p = 1 - W_2^2 / W_1^2$ で示す C_p が0.6を超えれ
ば必ず失速するとされている。筆者もこの限界に
挑戦したが翼断面形状や翼配列の工夫のみで此の
壁を越えることは出来なかった。此の発明は翼列部分の湾曲した振り通路に於
て、翼背側と壁面で構成される角に発生する低エ
ネルギー部に主流の持つ速度エネルギーの一部を補給
することにより、 $C_p = 0.6$ が限界とされた従来の
失速限界を超えることを目的とする。この発明を図面にもとづいて説明すれば次の通
りである。第1図に動翼1及び静翼2(両者合
せて翼列3)周辺の公知の主要構成を示す。翼列と
共に外側通路壁を構成する外車室4、内側通路壁
を構成する内車室5及びロータ6、又回転するロ
ータを支える軸受7よりなる。此の流体通路を構
成する翼列部分を拡大して壁面8、すなわち動翼

(1)

(2)

(1)にとっては外車室(4)、静翼(2)にとっては内車室(5)がある。一方翼根方向には翼をとりつけた壁面9、すなわち動翼(1)にとってはロータ(6)、静翼(2)にとっては外車室(4)がある。

第3図は翼列風洞試験で翼背面に生じる流跡を示す。壁面8に夾まれた翼列3の流入端10及び流出端11に対し、流入前の主流12は翼背面で13の如く加速され、壁面8に近い部分に2次流14を生じ、これが翼列失速の原因となる。

この発明は第4図に示す如く翼の壁面8に近い翼流入端10に突起15をとりつけることにより、主流の一部を壁面方向流16とし、この影響で2次流14を減少させたことである。尚此の突起15周辺を拡大すれば第5図の如く翼流入端10には平行な部分17と、これと翼流入端10を結ぶテーパー部18を設けることにより、2次流減少効果を更に増すことが出来た。

更にこの発明の効果として、2次流が第3図から第4図へと大巾に減少することにより、大巾な効率向上を果し、又翼流出端11における笛吹き作

(3)

用による騒音も、翼出口端11における主流13の流速が減じることによりそのエネルギーを半減する。

この発明は以上で説明した如く、従来の減速翼列の流入端で壁面に近い部分に突起をつけることにより、翼列の失速限界を上げ、効率を改善し、更に騒音を減少させる。

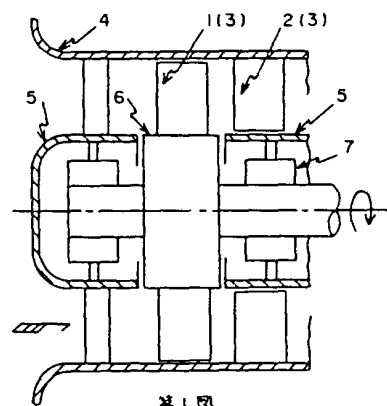
4. 図面の簡単な説明

図はこの発明の実施例を示すものである。第1図は公知の翼周辺主要構成の断面を示す。第2図は翼列部断面拡大図である。第3図は従来の翼、第4図はこの発明による突起つき翼の、それぞれ翼列風洞による翼背面上流跡の平面図を示す。第5図はこの発明の翼前縁にとりつけた突起部拡大平面図を示す。

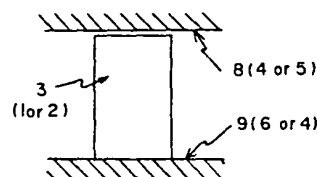
15……翼の壁面8に近い翼流入端10にとりつけた突起、17……突起15の内翼流入端10には平行な部分、18……この平行な部分17と翼流入端10を結ぶテーパー部

特許出願人 上松慎二

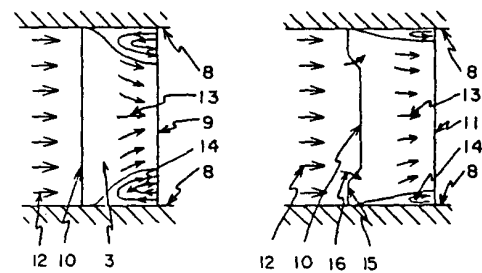
(4)



第1図

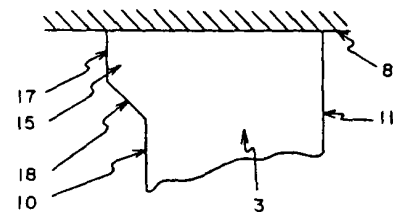


第2図



第3図

第4図



第5図

Controlling Secondary-Flow Structure by Leading-Edge Airfoil Fillet and Inlet Swirl to Reduce Aerodynamic Loss and Surface Heat Transfer

T. I-P. Shih and Y.-L. Lin

Department of Mechanical Engineering, Michigan State University
East Lansing, Michigan, U.S.A. 48824-1226

ABSTRACT

Computations, based on the ensemble-averaged compressible Navier-Stokes equations closed by the shear-stress transport (SST) turbulence model, were performed to investigate the effects of leading-edge airfoil fillet and inlet-swirl angle on the flow and heat transfer in a turbine-nozzle guide vane. Three fillet configurations were simulated: no fillet (baseline), a fillet whose thickness fades on the airfoil, and a fillet whose thickness fades on the endwall. For both fillets, the maximum height above the endwall is positioned along the stagnation zone/line on the airfoil under the condition of no swirl. For each configuration, three inlet swirls were investigated: no swirl (baseline) and two linearly varying swirl angle from one endwall to the other ($+30^\circ$ to -30° and -30° to $+30^\circ$).

Results obtained show that both leading-edge fillet and inlet swirl can reduce aerodynamic loss and surface heat transfer. For the conditions of this study, the difference in stagnation pressure from the nozzle's inlet to its exit were reduced by more than 40% with swirl or with fillet without swirl. Surface heat transfer was reduced by more than 10% on the airfoil and by more than 30% on the endwalls. When there is swirl, leading-edge fillets became less effective in reducing aerodynamic loss and surface heat transfer, because the fillets were not optimized for swirl angles imposed. Since the intensity and size of the cross flow were found to increase instead of decrease by inlet swirl and by the type of fillet geometries investigated, the results of this study indicate that the mechanisms responsible for aerodynamic loss and surface heat transfer are more complex than just the intensity and the magnitude of the secondary flows. This study shows their location and interaction with the main flow to be more important, and this could be exploited for positive results.

NOMENCLATURE

- C1 no airfoil fillet (Fig. 4)
- C2 fillet whose thickness fades on the airfoil (Fig. 4)
- C3 fillet whose thickness fades on the endwall (Fig. 4)
- C_x airfoil chord length projected on the X-coordinate
- h heat transfer coefficient ($h = q'' / (T_w - T_i)$)
- k turbulent kinetic energy

- M_x average inlet Mach number based on X-component velocity
- NS no swirl
- P static pressure or pitch (Fig. 2)
- P_{ref} reference static pressure (P_{ref} is pressure at nozzle exit)
- q'' surface heat transfer per unit area ($-\kappa \partial T / \partial n$)
- S1, S2 nozzle inlet/outlet height (Fig. 2); also swirl 1/2 (Fig. 4)
- T_w wall temperature
- T_i hot gas temperature at nozzle inlet
- ρ_i inlet averaged density
- U_i average X-component velocity of mainflow at nozzle inlet
- U_τ friction velocity of mainflow ($\sqrt{\tau_w / \rho_w}$)
- X, Y, Z coordinate direction defined in Fig. 1 ($X = 0$ at nozzle exit)
- y^+ normalized distance normal to wall ($\rho U_\tau Y / \mu$)
- Greek
- κ thermal conductivity
- ν kinematic viscosity
- ρ density
- θ swirl angle ($\cos^{-1} M_x / M_i$)
- ω dissipation rate per unit k
- Subscripts and Superscripts
- i inlet conditions
- w wall

INTRODUCTION

Secondary flows in airfoil passages such as horseshoe, passage, and corner vortices are generally considered detrimental to the performance and reliability of gas turbines (Langston, 2000). This is because secondary flow gets its energy from the main flow, and so is a source of aerodynamic loss. Also, they entrain higher-temperature gas in the main flow to the airfoils and the endwalls, and this can increase surface heat transfer. On film cooling, secondary flows can advect and lift film-cooling jets away from surfaces that they are intended to protect. These potentially adverse effects can be especially severe for low-aspect ratio, high-pressure turbines and highly loaded, low-pressure turbines.

A number of design concepts have been advanced to minimize the intensity and size of secondary flows. One promising concept is endwall contouring, especially the contouring, which accelerates the main flow upstream of the airfoil and continues the acceleration

throughout the airfoil passage (see e.g., Deich, et al. (1960), Ewen, et al. (1973), Dossena, et al. (1999), Duden, et al. (1999), Burd & Simon (2000), Shih, et al. (2000), and Lin, et al. (2000, 2001)). Though promising, this design concept can only be applied to the first-stage stator, where there is a contraction in cross-sectional area from the combustor to the turbine.

Another promising design concept is contouring the airfoil next to the endwalls, often referred to as airfoil fillets. This design concept has the advantage of being applicable to airfoil passages in any stage. Most of the research on airfoil fillets has been on wing/body junctures (see e.g., Pierce, et al. (1988), Davenport, et al. (1990), Pierce & Shin (1992), and references cited there). These studies were primarily concerned with the elimination of horseshoe vortices. They found that a pointed/sharp fillet on the leading edge of the airfoil (Fig. 1(a)) could reduce or eliminate horseshoe vortices. But, the "pointedness and sharpness" of the fillet must be aligned with the stagnation flow. For rounded fillets (Fig. 1(b)), they found the intensity and size of horseshoe vortices to increase with the radius of the fillet. They also found the horseshoe vortex structure to be displaced further upstream of the leading edge.

Very few studies have reported the use of airfoil fillets in gas turbines, where there are not only horseshoe-type vortices but also Dean-type vortices induced by the pressure difference between the pressure and suction sides of adjacent airfoils. Using leading-edge fillets to eliminate horseshoe vortices was recently proposed by Bancalari and Nordlund (1999), and then reiterated by Shih (2000). Zess and Thole (2001) reported a combined numerical and experimental study on leading-edge airfoil fillet, and found that the pointed/sharp-type of fillet shown in Fig. 1(a) can indeed eliminate horseshoe vortices. They also noted that the turbulent kinetic energy is greatly reduced in the endwall region. Sauer, et al. (2000) studied a different type of fillet, one that enhances instead of diminishes horseshoe vortices. Their fillet, a bulb-type (Fig. 1(c)), is intended to intensify horseshoe vortices on the suction side in order to reduce the pressure-side-to-suction-side secondary flow.

Thus, most research on secondary flows has been on how to suppress or minimize the intensity and size of secondary flows. In this study, that is not the objective. Instead, it is to investigate how secondary flows can be harnessed to produce positive results – even if secondary flows intensify. More specifically, the objective is to use computational fluid dynamics (CFD) to explore secondary flows induced by leading-edge airfoil fillet and inlet swirl in a nozzle guide vane and their effects on aerodynamic loss and surface heat transfer. Three leading-edge fillets are studied: no fillet (baseline), a fillet whose thickness fades on the airfoil, and a fillet whose thickness fades on the endwall. Since swirl can be quite high in magnitude at combustor exit (see, e.g., Goebel, et al. (1993) and van Fossen & Bunker (2001)), three inlet swirls were studied for each fillet configuration: no swirl (baseline) and two linearly varying swirl angle from one endwall to the other ($+30^\circ$ to -30° and -30° to $+30^\circ$). These swirls are taken to simulate flow conditions from can-type combustors with fuel/air swirlers.

PROBLEM DESCRIPTION

A schematic diagram of one periodic section of the nozzle guide vanes studied is shown in Fig. 2. A three-dimensional (3D) rendering is shown in Fig. 3. For this linear cascade, the airfoil is two-dimensional (2D) with an axial chord length C_x of 19.883 cm.

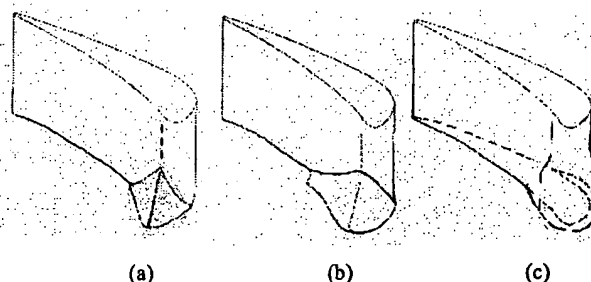


Fig. 1. Basic leading-edge fillet geometries. (a) Sharp/pointed. (b) Rounded. (c) Bulb-type.

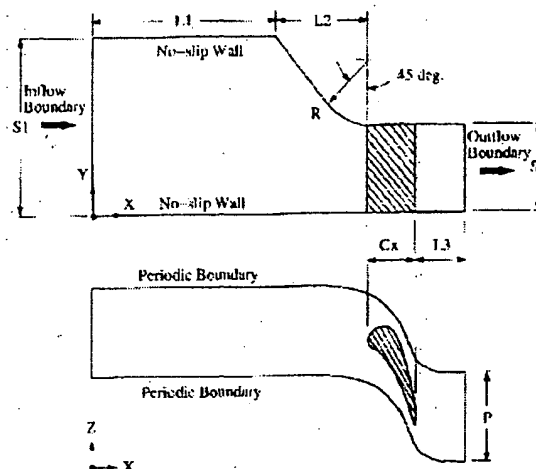


Fig. 2. Schematic of the nozzle vane studied (not drawn to scale).



Fig. 3. Three-dimensional rendering of the nozzle vane studied.

The actual chord length C is 31.8 cm. The pitch P between airfoils is 31.8 cm. On the endwalls, one is flat, and another is contoured. For the contoured endwall, the profile is made up of straight line with a 45° slope and a 45° arc with radius R of 24.41 cm. Note that all of the contouring is upstream of the airfoil (Figs. 2 and 3). This is the more traditional design in which contouring is not utilized to modify the flow within the airfoil passage. The distance from the inlet to the beginning of contouring $L1$ is 58.4 cm; the distance of the contouring $L2$ is 35.8 cm; and the distance from the airfoil's trailing edge to the outlet $L3$ is 23.8 cm. The nozzle has an inlet height $S1$ of 44.45 cm and an outlet height $S2$ of 24.41 cm.

For this 2D airfoil, three different leading-edge fillets were studied (Fig. 4): no fillet (baseline; referred to as C1

configuration), a fillet whose thickness diminishes on the airfoil surface (C2 configuration), and a fillet whose thickness diminishes on the endwall (C3 configuration). For both fillets, the maximum height above the endwall is positioned along the stagnation zone/line on the airfoil for the case with no swirl. Note that the two fillets studied are of the rounded type (Fig. 1(b)), which are known to increase horseshoe vortices instead of reducing them.

The flow through the nozzle is air. At the nozzle inlet, the air temperature is uniform at $T_i = 300$ K. The X-component (stream-wise) velocity at the nozzle inlet has a one-seventh power-law profile. Though this profile may not be the most realistic, it is no worse than any other profile in serving as a reference. In an actual combustor, the velocity profile at the turbine inlet can be incredibly complicated with large-scale turbulence structures, depending on the combustor design and operation. In this study, the averaged inlet Mach numbers based on the X-component velocity M_x is 0.05. With an inlet temperature T_i of 300 K, the corresponding averaged X-component velocity U_i is 17.4 m/s, which give rise to a Reynolds numbers (Re) based on C_x of 438,000. The turbulence intensity and the turbulent viscosity at the nozzle inlet are taken to be 10% of U_i and 100 times the laminar viscosity, respectively (Burd & Simon (2000)). The X-component velocity at the inflow boundary is kept constant when swirl is added.

In this study, swirl is introduced at the nozzle inlet by setting the Y-component velocity v to zero, by keeping the X-component velocity u the same as the no swirl case, and by adjusting the Z-component velocity w to produce the desired swirl angle ($\theta = \tan^{-1} w/u$). Three swirls were simulated (Fig. 5): no swirl (baseline; referred to as NS), one linearly varying swirl angle with $\theta = +30^\circ$ on the upper contoured endwall and -30° on the lower flat endwall (S1 swirl), and one linearly varying swirl angle with $\theta = -30^\circ$ on the upper endwall and $+30^\circ$ on the lower endwall (S2 swirl). The S1 swirl (Figs. 5(b) and 5(c)) produces positive angles of attack on the airfoil from mid-span to the upper endwall, and negative angles of attack from the mid-span to the lower endwall. The S2 swirl produces negative angles of attack on the airfoil from mid-span to the upper endwall, and positive angles of attack from the mid-span to the lower endwall.

All surfaces of the airfoil and endwalls are maintained at a constant temperature T_w of 270 K. The average static pressure at the nozzle outlet is maintained at 1 atm ($P_{ref} = 101325$ Pa). A summary of the cases studied is given in Table 1.

PROBLEM FORMULATION

The flow and heat transfer in the nozzle guide vane described in the previous section are modeled by the ensemble-averaged conservation equations of mass (continuity), momentum (compressible Navier-Stokes), and total energy for a thermally and calorically perfect gas with Sutherland's model for viscosity and a constant Prandtl number. These ensemble-averaged equations were closed by the Shear-Stress-Transport (SST) turbulence model (Menter (1991, 1993)). The SST model is a blend of the $k-\omega$ model in the near-wall region and the $k-\epsilon$ model further away from the wall. Though the SST model uses the $k-\omega$ model in the near-wall region, dependence on freestream k has been eliminated. Also, a limiter is introduced to control overshoot in k with adverse pressure gradients so that separation is predicted more accurately.

The SST model does not require explicit damping in the near-

wall region because ω becomes very large as the wall is approached. In this study, integration of the conservation equations as well as the turbulence model is made all the way to the wall (i.e., wall functions are not used). The boundary conditions (BCs) used on all walls are no-slip (zero velocity), isothermal walls (constant T_w), and zero turbulent kinetic energy. The dissipation rate per turbulent kinetic energy, ω , on the wall is set to $60\nu/\beta \xi^2$, where β equals 3/40 and ξ is the normal distance of the first grid point from the wall in which that first point must be within a y^+ of one. Other BCs are as follows (see Fig. 2): At the inflow (nozzle inlet), the velocity, temperature, k , and ω are specified (see Problem Description). At the outflow (nozzle outlet), the average static pressure is specified, but the pressure gradients in the Y and Z directions are extrapolated. Also extrapolated are density, velocity, k , and ω . At the two planes denoted as periodic boundaries, periodic boundary conditions are imposed.

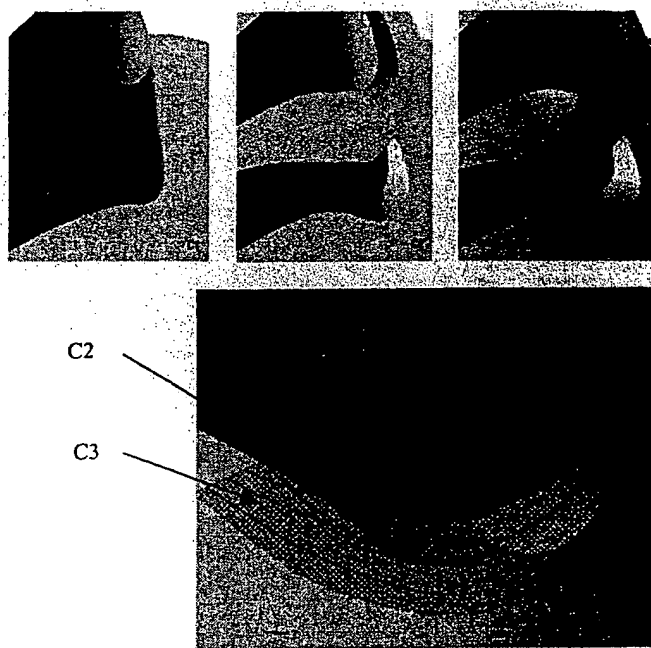


Fig. 4. Fillet configurations studied. C1: no fillet. C2: merge on airfoil. C3: merge on endwall.

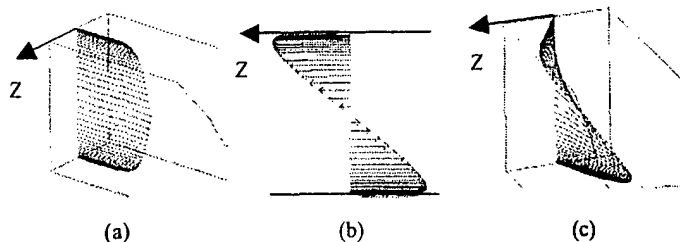


Fig. 5. Illustration of swirls investigated. (a) No swirl with $1/7^{th}$ profile. (b) S1 swirl (2D view). (c) S1 swirl (3D view). S2 is just the opposite of S1 and so is not shown.

Table 1. Summary of Cases Studied*

Case No.	Fillet Configuration	Swirl Angle (degrees)
C1-NS	no fillet	no swirl
C1-S1	no fillet	S1: +30 to -30
C1-S2	no fillet	S2: -30 to +30
C2-NS	merge on airfoil	no swirl
C2-S1	merge on airfoil	S1: +30 to -30
C2-S2	merge on airfoil	S2: -30 to +30
C3-NS	merge on endwall	no swirl
C3-S1	merge on endwall	S1: +30 to -30
C3-S2	merge on endwall	S2: -30 to +30

* $M_x = 0.05$, $T_i = 300K$, $T_w = 270 K$, $P_{ref} = 101325 Pa$.

NUMERICAL METHOD OF SOLUTION

Solutions to the conservation equations were obtained by using a cell-centered finite-volume code called CFL3D (see Thomas, et al. (1990) and Rumsey & Vatsa (1993)). This research code, developed at NASA - Langley Research Center, is widely used in the aerodynamics community, and has been validated by the authors for gas-turbine heat transfer studies (see, e.g., Lin, et al. (2001), Lin & Shih (2001) and Shih & Sultanian (2001)). In this code, all inviscid terms are approximated by the third-order flux-difference splitting of Roe (1986) with limiters. Flux-difference was used so that numerical diffusion would be much smaller than physical diffusion. All diffusion terms are approximated conservatively by differencing derivatives at cell faces. Since only steady-state solutions are of interest, time derivatives were approximated by the Euler implicit formula. The systems of nonlinear equations that resulted from the aforementioned approximations to the space- and time-derivatives were analyzed by using a diagonalized alternating-direction implicit scheme (Pulliam & Chaussee (1981)) with local time-stepping and three-level V-cycle multigrid (Anderson et al. (1988)).

The domain of the nozzle-guide vane investigated (region within the solid lines in Fig. 2) is represented by a multi-block structured grid system made up of five H-H grids. Figure 6 shows the overall grid for the case without fillets, and Fig. 7 shows the grids with the fillets. The number of grid points used is 1,658,685 for all cases summarized in Table 1. The number and distribution of the grid points used were obtained by satisfying a set of rules established by developers of numerical methods and turbulence models to ensure accuracy. These rules include having at least five grid points within a y^+ of 5 with the first point within a y^+ of unity and having constant grid spacing next to walls for at least three grid points to resolve turbulent boundary layers. Other rules include aligning grid lines with flow direction as much as possible and keeping grid-aspect ratio near unity in regions with recirculating flow. As a further test, grids were refined by a factor of 25% in each direction for Cases C1-NS, C2-NS, and C3-NS. This test showed the predicted pressure difference in the stagnation pressure from inflow to outflow boundary normalized by the stagnation pressure at the inflow boundary to vary by less than 2% when compared to the more refined grids.

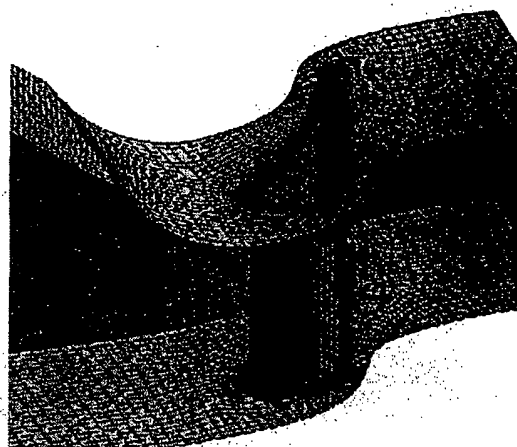


Fig. 6. Multiblock grid system used. Not all grid lines shown.

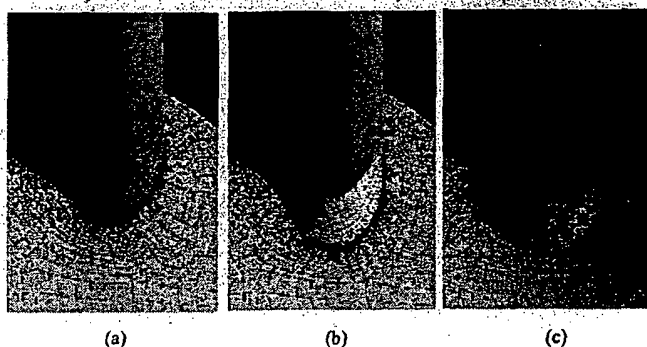


Fig. 7. Grid system about the airfoil/flat-endwall junction. Not all grid lines shown. (a) C1. (b) C2. (c) C3.

RESULTS AND DISCUSSION

The objective of this study is to use CFD to investigate secondary flows induced by leading-edge airfoil fillet and inlet swirl and how they affect aerodynamic loss and surface heat transfer in a nozzle guide vane. The goal is not to minimize secondary flows, but rather to see if secondary flows can be harnessed in a positive way. To accomplish this, nine CFD simulations were performed (see Table 1). Results of these simulations are summarized in Table 2 and displayed in Figs. 8 to 14. In the following, we first examine if fillets and swirl can affect aerodynamic loss and surface heat transfer in a positive way. Afterwards, we examine the mechanisms involved.

Aerodynamic Loss and Surface Heat Transfer

Table 2 summarizes a number of integrated results obtained from all simulations performed: the change in the average stagnation pressure from the nozzle inflow boundary to the nozzle outflow boundary ($\Delta P_{stagnation}$), the net pressure force acting on the airfoil, the net surface heat transfer rate (Q) on the airfoil and the two endwalls, and the percent change in mass flow rate through the

Table 2. Summary of Aerodynamic Loss, Surface Heat Transfer, and Change in Mass Flow Rate*

Case No.	$\Delta P_{\text{stagnation}}$ (Pa)	Q_{contour} (W)	Q_{flat} (W)	Q_{airfoil} (W)	$\frac{\Delta \dot{m}}{\dot{m}}$
C1-NS	3006.2	3104.6	3075.1	2214.9	0.00%
C1-S1	1707.8	2076.9	2033.0	1948.8	1.74%
C1-S2	1668.8	1697.1	1727.3	1804.4	1.92%
C2-NS	1714.2	2083.7	2077.9	1909.6	1.74%
C2-S1	1865.2	1843.2	1773.2	1832.7	1.87%
C2-S2	1718.8	1830.7	1811.5	1819.1	1.87%
C3-NS	1688.7	1976.5	1953.6	1847.6	1.81%
C3-S1	1682.4	1803.0	1708.0	1810.0	1.90%
C3-S2	1706.3	1767.4	1763.5	1792.6	1.90%

* $\Delta P_{\text{stagnation}}$ is the change in stagnation pressure from nozzle inlet to exit (Fig. 2). Q_i is the net heat transfer rate from surface i (i denoted airfoil, contoured & flat endwall). $\Delta \dot{m}/\dot{m} = (\dot{m} - \dot{m}_{\text{C1-NS}})/\dot{m}_{\text{C1-NS}}$ is increase in mass flow rate relative to baseline case C1-NS.

nozzle guide vane relative to the baseline case of C1-NS (i.e., $\Delta \dot{m}/\dot{m} = (\dot{m} - \dot{m}_{\text{C1-NS}})/\dot{m}_{\text{C1-NS}}$). $\Delta P_{\text{stagnation}}$ is taken to be a measure of the aerodynamic loss. From this table, the effects of inlet swirl and leading-edge fillets on aerodynamic loss and surface heat transfer can be assessed.

The effects of inlet swirl without the complication of fillets can be evaluated by comparing C1-NS (baseline: no fillets and no swirl) with C1-S1 (no fillets, S1 swirl) and C1-S2 (no fillets, S2 swirl). This comparison shows that when swirl is added, $\Delta P_{\text{stagnation}}$ is reduced by 43.2% for S1 and by 44.5% for S2. Surface heat transfer is also greatly reduced by swirl. On the airfoil, Q is reduced by 12.0% for S1 and 18.5% for S2. On the endwalls, the reduction in Q is even more, 33.5% for S1 and 44.6% for S2. Thus, both S1 and S2 swirls are quite effective in reducing aerodynamic loss and surface heat transfer. Though aerodynamic loss is greatly reduced with S1 and S2 swirls, the increase in mass-flow rate through the nozzle-guide vane is quite small, less than 2%. This is because the boundary layers next to solid surfaces account for a small fraction of the flow-path cross-section. Note that the results for S1 and S2 differ because of the contouring of one endwall upstream of the airfoil. Basically, if the contouring is on the top wall, then it is more advantage to have a swirl that produces a negative angle of attack for that endwall.

The effects of leading-edge fillets on aerodynamic loss and surface heat transfer can be evaluated by comparing C1-NS (baseline: no fillets and no swirl) with C2-NS (C2 fillet, no swirl) and C3-NS (C3 fillet, no swirl). This comparison shows that $\Delta P_{\text{stagnation}}$ is reduced by 42.9% for the C2 fillet and by 43.8% for the C3 fillet. For surface heat transfer on the airfoil, it is reduced by 13.8% for the C2 fillet and 16.6% for the C3 fillet. The reduction in Q on the endwalls is even more, 32.7% for C2 fillet and 36.4% for C3 fillet. Again, though aerodynamic loss decreases considerably, increase in mass flow rate is less than 2%.

Thus, it can be seen that both leading-edge airfoil fillet and inlet swirl significantly reduce aerodynamic loss and surface

heat transfer on its own. The effects of having both swirl and fillet can also be evaluated. By comparing, C2-S1 and C3-S1 with C1-S1, we see the C2 fillet tends to increase and the C3 fillet tends to decrease aerodynamic loss slightly. Both C2 and C3 fillets further reduced surface heat transfer with C3 fillet reducing slightly more. By comparing, C2-S2 and C3-S2 with C1-S2, we see both C2 and C3 fillets tend to increase instead of decrease aerodynamic loss and surface heat transfer by a small percentage. The poorer performance of C2 and C3 fillets for S2 swirl may indicate that fillet shape needs to be optimized with respect to swirl. Since the reduction in aerodynamics loss and surface heat transfer is small when we have both swirl and fillets,

Induced Secondary Flows

In this section, we examine the size and intensity of the secondary flows induced by inlet swirl and leading-edge fillets.

Since a reduction in aerodynamic loss and surface heat transfer for a given airfoil configuration/passage is often attributed to reduced secondary flows, we examine the secondary flows induced by inlet swirl and fillets. Figure 8 shows the cross-flow velocity vector and magnitude in an A-A plane perpendicular to the airfoil flow passage near the trailing edge of the pressure surface. From part (a) of this figure, the following can be observed. For the case without fillet and without swirl (C1-NS), the cross flow is asymmetric because of the contouring on the upper endwall. By adding S1 swirl (which produces greater positive angle of attack at the upper endwall), this asymmetry is greatly enhanced with a corresponding increase in size and intensity of the cross flow. With the S2 swirl (which produces a negative angle of attack at the upper endwall), the asymmetry is reversed with the cross flow being larger near the lower endwall (denoted as flat) instead of the upper endwall (denoted as contoured). Though S2 swirl produces considerably less cross flow than S1 swirl, both S1 and S2 produce more cross flow than the case without swirl.

By comparing C2-NS and C3-NS with C1-NS in parts (a), (b), and (c) in Fig. 8, it can be seen that adding fillets also does not reduce the cross flow. Cross flow size and magnitude increased slightly with the C2 fillet and decreased slightly with the C3 fillet. If swirl is added with the fillets, then cross flows are enhanced but less than when there are no fillets.

Figure 9 shows streamlines approaching the airfoil's leading edge near the airfoil/flat-endwall junction. From that figure, it can be seen that the fillets investigated do not eliminate or minimize horseshoe vortices. Not shown is that the swirls investigated also do not eliminate or reduce horseshoe vortices. Both swirl and fillet tend to push the horseshoe-vortex structure further upstream of the airfoil's leading edge. This is consistent with the observations made by previous investigators (e.g., Pierce, et al. (1988), Devenport, et al. (1990), Pierce & Shin (1992)).

Since horseshoe vortices at the airfoil's leading edge/endwall junction and cross flows at the airfoil's trailing edge were not diminished and yet aerodynamic loss and surface heat transfer are reduced significantly, this must mean that the mechanism responsible for reduction is much more complex than just the size and intensity of secondary flows. The fact that the size and intensity of secondary flow alone do not tell the whole story should not be unexpected. This is because the streamwise component of the velocity is so much greater than the cross-flow components. But, secondary flows, especially those near the airfoil/endwall

junctions could affect the flow structure and the evolution of the boundary layers, which do affect significantly the aerodynamic loss and surface heat transfer.

Flow-Structure Near Endwalls and Airfoil Surface

In this section, we examine the flow structure near the airfoil/endwall junction with and without swirl and with and without fillets. Figure 10 shows the pressure contours on the airfoil/flat endwall and the velocity vectors very close to those surfaces (y^+ between 10 and 20). From this figure, it can be seen that both swirl and fillets increase the pressure and the size of the high-pressure (stagnation) region on the endwall about the airfoil's leading edge. When there is no fillet, even S1 swirl with negative angle of attack increases the pressure there. With S2 swirl, which provides a positive angle of attack at the flat endwall, the increase is even more substantial. The imposed S1 and S2 swirls increase the magnitude and the size of the high-pressure region on the endwalls about the airfoil's leading edge because they increase the kinetic energy of the approaching flow, especially in the region close to the endwall (see Fig. 5). The fillets investigated increase the extent of the stagnation region on the endwalls by geometrically extending the airfoil forward. This increased pressure, whether from swirl or from the fillet, is why horseshoe vortices form further upstream of the airfoil. Figure 11 shows how swirl affects the streamlines very close to the endwalls. From this figure, the extent of the upstream influence of the airfoils's leading edge can clearly be seen. The increased size of the stagnation zones on the endwalls about the airfoil's leading edge lowers the flow speed and velocity gradients there, which in turns reduces turbulence production. This could be one of the main reasons for the reduced aerodynamic loss and surface heat transfer on the endwall. However, there may be other reasons or a point of diminishing returns. This is because Fig. 10 shows C2-S1, C2-S2, C3-NS, C3-S1, and C3-S2 to have larger regions of higher pressure on the endwall than those from C1-S1 or C2-NS, but have comparable aerodynamic loss and surface pressure.

On the airfoil surface, the swirls and fillets investigated shift the location of maximum pressure on the airfoil surface further downstream of the geometric leading edge near the endwalls. Moving the location of the stagnation zone downstream of the geometric leading edge, where the airfoil is flatter, can be significant. This is because the boundary-layer thickness at the stagnation zone is finite (> 0), whose value depends on the surface curvature. As a first approximation, that thickness is $2.4\sqrt{\nu/\lambda}$ from the similarity solution of the Hiemenz flow, where ν is the effective kinematic viscosity and λ depends on surface curvature. If the surface is flat, then $\lambda = U_\infty$, where U_∞ is the speed of the approaching flow far upstream of the surface. For a cylindrical surface of radius r_0 , λ can be approximated by $2U_\infty/r_0$. Thus, the flatter the surface, the larger is the boundary-layer thickness at the stagnation zone. With a larger boundary-layer thickness, the boundary-layer development on the airfoil from the stagnation zone will be different since the gradients will be less. This could explain why aerodynamic loss is reduced on the airfoil.

When there is swirl, Fig. 8 shows the cross flow from the pressure to the suction surface reaches the suction surface if the angle of attack from swirl is positive near that endwall. This is because when the angle of attack is positive, static pressure on the

pressure surface near the endwall is increased, which increases the pressure difference between the pressure and suction surfaces. If the angle of attack is negative near an endwall, then Fig. 8 shows the cross flow to not reach the suction surface. This is because when the angle of attack is negative, the pressure on the suction surface near the endwall is increased. This increased pressure diverts flow away from the suction surface, causing the cross flow from the pressure surface to lift off from the endwall before reaching the suction surface. This can be seen in Figs. 8 and 11.

Distribution of Heat Transfer Coefficient

Figures 12 to 14 show the computed heat transfer coefficient on the airfoil and the endwall surfaces. When there is no swirl and fillets, Fig. 12 (C1-NS) shows that the heat-transfer coefficient is high on the airfoil's suction surface near the blade-to-blade throat region (i.e., where the normal distance between blades is the minimum) because of the high-speed flow there. Heat transfer rate is also high on the parts of the suction surface airfoil next to the endwalls because of the small-scale secondary flows there, transporting hotter fluid from the mainstream to the airfoil surface. On the endwalls, the heat transfer rate is high everywhere except about the stagnation region next to the pressure surface of the airfoil, where the velocities near the surface have been reduced. The above explanations are the traditional ones, which apply for the case without swirl and fillets.

When there is swirl and fillet, Figs. 12 to 14 show that the locations of where heat transfer rate (q) are high and low are similar to the case without swirl and fillets with two exceptions. The first is the reduced size of the region with higher q , and the second is markedly reduced magnitudes of higher q . These differences can be linked to the secondary flow structures induced by swirl and fillets.

When there is swirl, fillet, or both, Figs. 12 to 14 show q to be reduced on the endwall about the airfoil's leading edge because of the increased size of the high-pressure (stagnation) region, which produced lower flow speeds and gradients. The maximum q on the airfoil's suction surface near the blade-to-blade throat region is reduced as well. Either swirl or fillet was also found to reduce q on the airfoil's pressure surface near the trailing edge.

When there is only the fillet and no swirl, q is still high on the parts of the suction surface airfoil next to the endwalls because of the small-scale secondary flows there, which is similar to the case without swirl and fillet (compare C1-NS with C2-NS and C3-NS). With swirl, q there can be reduced considerably, especially on the suction surface next to the flat (lower) endwall (compare C1-NS with C1-S1 and C1-S2).

When there is swirl and no fillet, Fig. 12 (C1-S1 and C1-S2) shows q to be higher on the lower or upper part of the airfoil suction surface depending upon S1 or S2 swirl. Also, one endwall has lower q than the other. The asymmetric heat transfer coefficient on the airfoil surface and endwalls is a clear sign of the effects of a large-scale secondary flow with a continuous versus a broken thermal boundary layer along the airfoil and endwalls. Swirl also reduces cross flow on one endwall.

SUMMARY

Computations performed show that inlet swirl and leading-edge fillet can reduce significantly aerodynamic loss and surface

heat transfer. This study shows that the size and the intensity of the secondary flows alone do not tell the whole story about aerodynamic and heat transfer. The swirls and fillets investigated were found to increase the size of the stagnation region on the endwall about the airfoil's leading edge. It also modified the stagnation region on the airfoil pressure surface and the boundary development from there. Fillets accomplish this geometrically. Swirl accomplishes this by increasing kinetic energy near the endwall region and by modifying the angles of attack.

ACKNOWLEDGEMENT

This research was supported by the U.S. Dept. of Energy's AGTSR Program with Dr. R. Wenglarz and Dr. L.P. Golan as the Technical Monitors. The authors are grateful for this support.

REFERENCES

- Anderson, W.K., Thomas, J.L., and Whitfield, D.L., 1988, "Multigrid Acceleration of the Flux-Split Euler Equations," *AIAA J.*, Vol. 26, No. 6, pp. 649-654.
- Bancalari, E. and Nordlund, S., DOE AGTSR Aero-Heat Transfer Workshop III, Austin, Texas, February 10-12, 1999.
- Burd, S.W. and Simon, T.W., 2000, "Flow Measurements in a Nozzle Guide Vane Passage with a Low Aspect Ratio and Endwall Contouring," ASME Paper 2000-GT-0213.
- Deich, M.E., Zaryankin, A.E., Fillipov, G.A., and Zatspin, M.F., "Method of Increasing the Efficiency of Turbine Stages and Short Blades," *Teploenergetika*, No. 2, Transaction No. 2816, Associated Electrical Industries (Manchester) Ltd., April 1960.
- Devenport, W.J., Agarwal, N.K., Dewitz, M.B., Simpson, R.L., and Poddar, K., "Effects of a Fillet on the Flow Past a Wing-Body Junction," *AIAA Journal*, Vol. 28, No. 12, 1990, pp. 2017-2024.
- Dossena, V., Perdichizzi, A., and Savini, M., "The Influence of Endwall contouring on the Performance of a Turbine Nozzle Guide Vane," *ASME J. of Turbomachinery*, Vol. 121, 1999, pp. 200-208.
- Duden, A., Raab, I., and Fottner, L., "Controlling the Secondary Flow in Turbine Cascade by Three-Dimensional Airfoil Design and Endwall Contouring," *ASME J. of Turbomachinery*, Vol. 121, 1999, pp. 191-199.
- Ewen, J.S., Huber, F.W., and Mitchell, J.P., "Investigation of the Aerodynamic Performance of Small Axial Turbines," ASME Paper 73-GT-3, 1973.
- Goebel, S.G., Abuaf, N., Lovett, J.A., and Lee, C.P., "Measurements of Combustor Velocity and Turbulence Profiles", Paper 93-GT-228, June 1993.
- Langston, L.S., 2000, "Secondary Flows in Axial Turbines - A Review," International Symposium on Heat Transfer in Gas Turbine Systems.
- Lin, Y.-L., Shih, T.I-P., Chyu, M.K., and Bunker, R.S., "Effects of Gap Leakage on Fluid Flow in a Contoured Turbine Nozzle Guide Vane," ASME Paper 2000-GT-0555, May 2000.
- Lin, Y.-L., Shih, T.I-P., Stephens, M.A., and Chyu, M.K., "A Numerical Study of Flow and Heat Transfer in a Smooth and a Ribbed U-Duct with and without Rotation," *ASME J. of Heat Transfer*, Vol. 123, April 2001, pp. 219-232.
- Lin, Y.-L. and Shih, T.I-P., "Film Cooling of a Semi-Cylindrical Leading Edge with Injection through Rows of Compound-Angled Holes," *ASME Journal of Heat Transfer*, Vol. 123, August 2001, pp. 645-654.
- Lin, Y.-L., Schock, H.J., Shih, T.I-P., and Bunker, R.S., "Effects of Inlet Conditions on Flow and Heat Transfer in Contoured Turbine Nozzle Guide Vanes," ASME Paper IMECE2001/HTD-24312, November 2001.
- Menter, F.R., 1991, "Performance of Popular Turbulence Models for Attached and Separated Adverse Pressure Gradient Flows," *AIAA J.*, Vol. 30, No. 8, pp. 2066-2071.
- Menter, F.R., 1993, "Zonal Two-Equation k- ω Turbulence Models for Aerodynamic Flows," AIAA Paper 93-2906.
- Pierce, E.J., Frangistas, G.A., and Nelson, D.J., "Geometry-Modification Effects on a Junction-Vortex Flow," *Proceedings of the Symposium on Hydrodynamics Performance Enhancement for Marine Applications*, October 1988, pp. 37-44.
- Pierce, E.J. and Shin, J., "The Development of a Turbulent Junction Vortex System," *ASME Journal of Fluids Engineering*, Vol. 114, 1992, pp. 559-565.
- Pulliam, W.R. and Chaussee, D.S., 1981, "A Diagonal Form of an Implicit Approximate Factorization Algorithm," *J. Computational Physics*, Vol. 39, pp. 347-363.
- Roe, P.L., 1986, "Characteristic Based Schemes for the Euler Equations," *Ann. Review of Fluid Mechanics*, Vol. 18, pp. 337-65.
- Rumsey, C.L. and Vatsa, V.N. 1993, "A Comparison of the Predictive Capabilities of Several Turbulence Models Using Upwind and Central-Difference Computer Codes," AIAA Paper 93-0192.
- Sauer, H., Mueller, R., Vogeler, K., "Reduction of Secondary Flow Losses in Turbine Cascades by Leading Edge Modifications at the Endwall," ASME Paper 2000-GT-0473, May 2000.
- Shih, T.I-P., DOE AGTSR Aero-Heat Transfer Workshop IV, Minneapolis, Minnesota, October 11-13, 2000.
- Shih, T. I-P., Lin, Y.-L., and Simon, T.W., "Control of Secondary Flow in Turbine Nozzle Guide Vane by Endwall Contouring," ASME Paper 2000-GT-0556, May 2000.
- Shih, T.I-P. and Sultanian, B., "Computations of Internal and Film Cooling," *Heat Transfer in Gas Turbines*, Editors: B. Suden and M. Faghri, WIT Press, Ashurst, Southampton, 2001, Chapter 5, pp. 175-225.
- Thomas, J.L., Krist, S.T., and Anderson, W.K., 1990, "Navier-Stokes Computations of Vortical Flows over Low-Aspect-Ratio Wings," *AIAA J.*, Vol. 28, No. 2, pp. 205-212.
- van Fossen, G.J. and Bunker, R.S., "Augmentation of Stagnation Region Heat Transfer Due to Turbulence from a DLN Can Combustor," *ASME J. of Turbomachinery*, Vol. 123, January 2001, pp. 140-146.
- Zess, G.A. and Thole, K.A., "Computational Design and Experimental Evaluation of Using a Leading Edge Fillet on Gas Turbine Vane," ASME Paper 2001-GT-0404, June 2001.

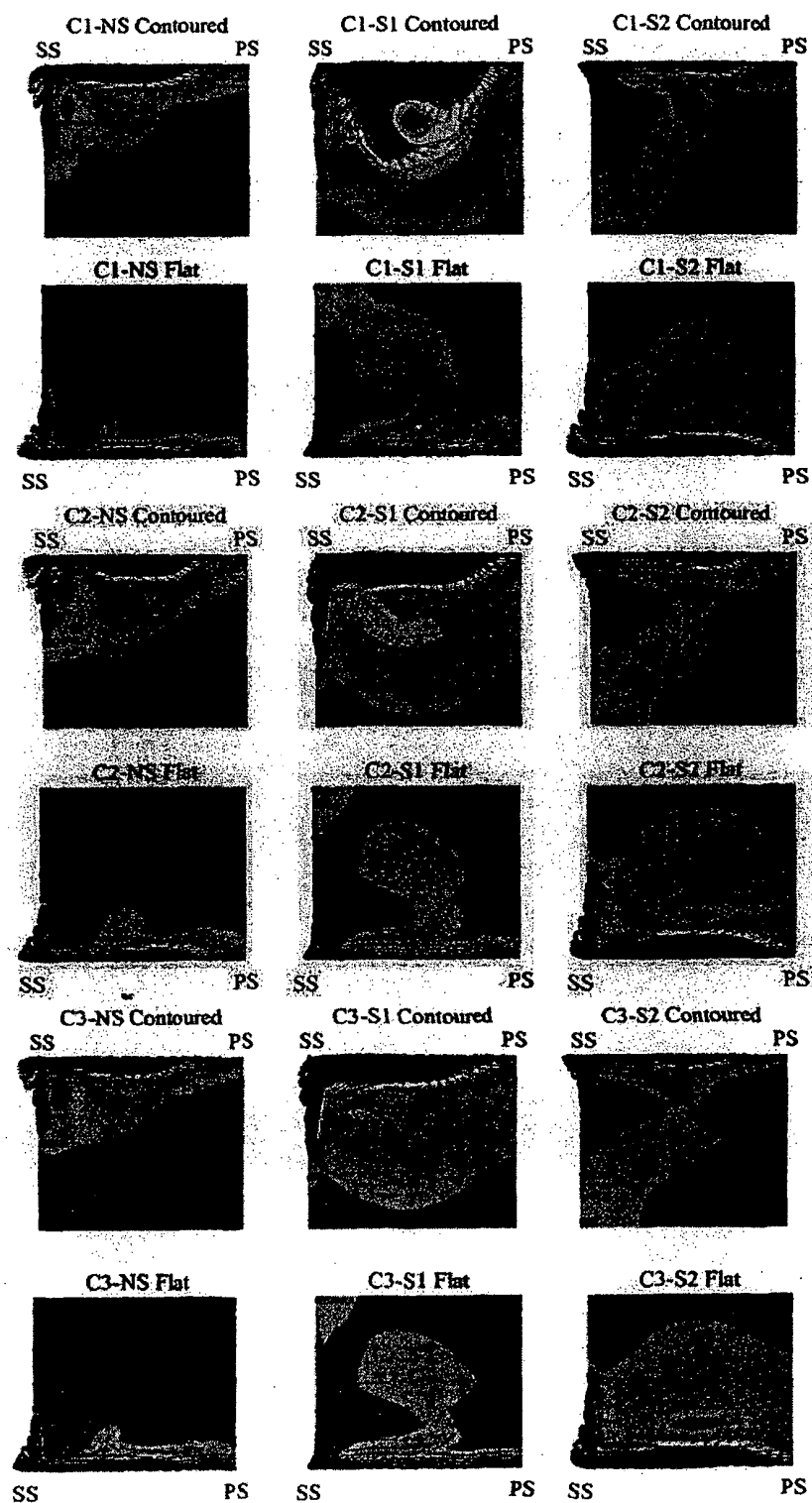


Fig. 8. Cross flow velocity vector and magnitude in plane A-A.



Fig. 9. Streamlines approaching the airfoil/flat-endwall junction, showing the generation of horseshoe vortices.

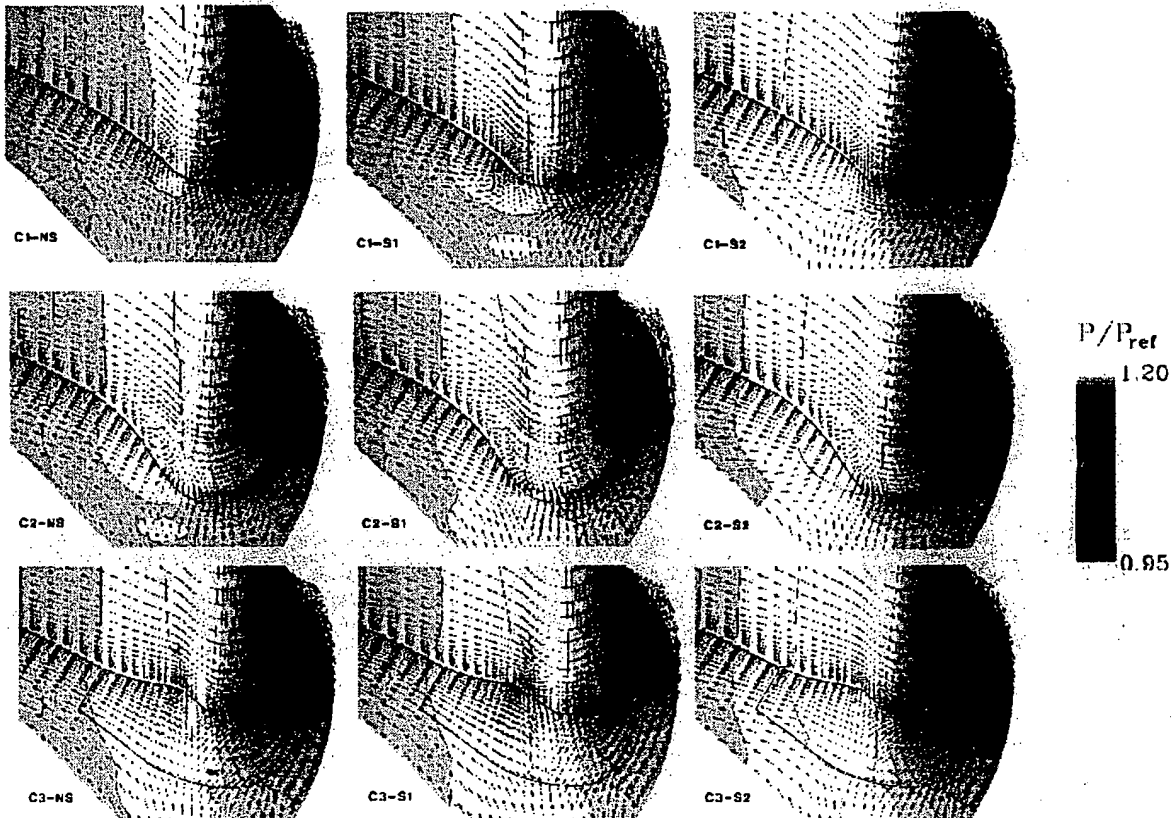


Fig. 10. Surface pressure and velocity vector near surface (y^+ between 10 and 20) at airfoil/flat-endwall junction.

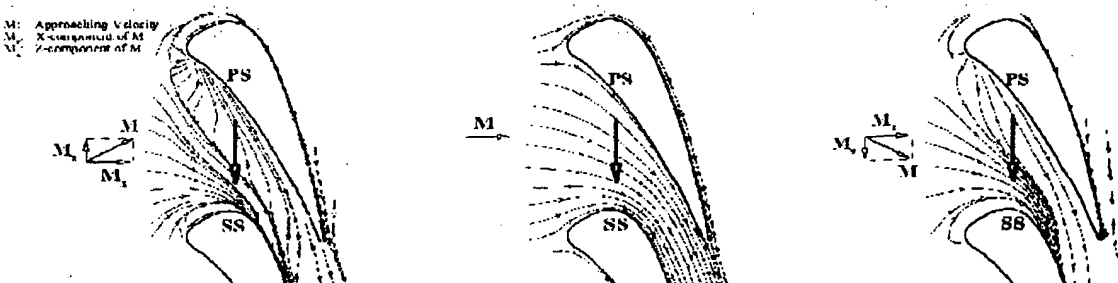


Fig. 11. Streamlines near the endwalls and the mid-plane with S1 swirl (no fillets).
(a) contoured endwall. (b) mid-plane. (c) flat endwall.

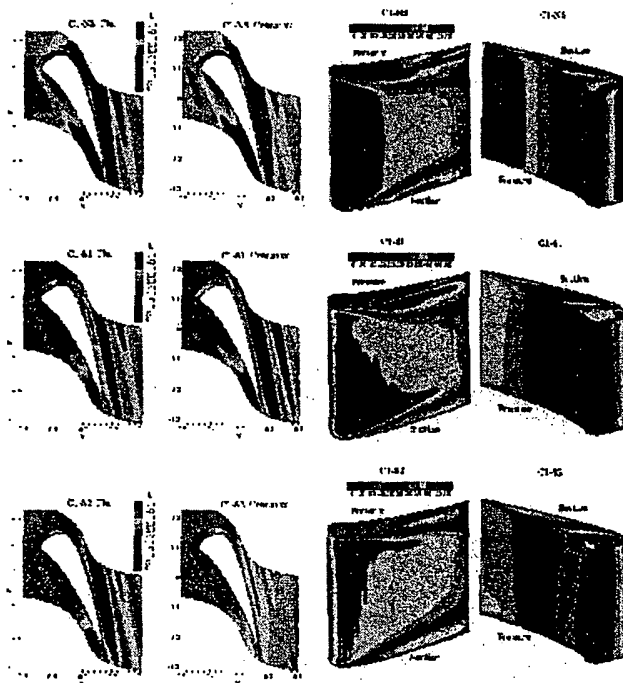


Fig. 12. Heat transfer coefficient: no fillet with and without swirl.

Fig. 13. Heat Transfer coefficient: C2 fillet with and without swirl.

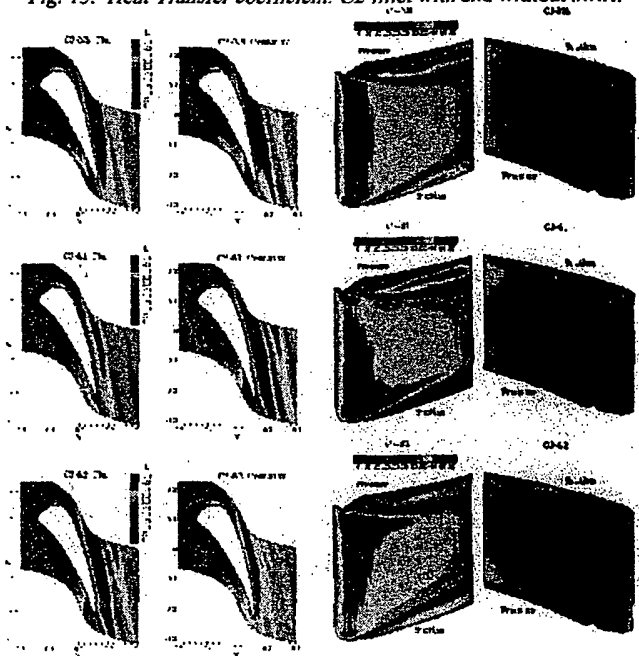


Fig. 14. Heat Transfer coefficient: C3 fillet with and without swirl.

